NASA/TM—2000-209933 ASME 2000–GT–0573

On Flowfield Periodicity in the NASA Transonic Flutter Cascade, Part II—Numerical Study

Rodrick V. Chima, Eric R. McFarland, and Jerry R. Wood
Glenn Research Center, Cleveland, Ohio

Jan Lepicovsky
Dynacs Engineering Company, Inc., Brook Park, Ohio

March 2000

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.
- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.
- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.
- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA Access Help Desk at (301) 621-0134
- Telephone the NASA Access Help Desk at (301) 621-0390
- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

NASA/TM—2000-209933 ASME 2000–GT–0573

On Flowfield Periodicity in the NASA Transonic Flutter Cascade, Part II—Numerical Study

Rodrick V. Chima, Eric R. McFarland, and Jerry R. Wood
Glenn Research Center, Cleveland, Ohio

Jan Lepicovsky
Dynacs Engineering Company, Inc., Brook Park, Ohio

Prepared for the
45th International Gas Turbine and Aeroengine Technical Congress
sponsored by the American Society of Mechanical Engineers
Munich, Germany, May 8–11, 2000

National Aeronautics and
Space Administration

Glenn Research Center

March 2000

Acknowledgments

The authors would like to acknowledge the invaluable engineering and technical support provided by Thomas Jett and Ricardo Torres. This program was supported by NASA Glenn Research Center, originally under the Smart Green Engine Program managed by Kestutis Civinskas, and currently under the Turbomachinery Combustion Technology Program managed by Robert Corrigan. Development of the PCSTAGE-TecPlot© interface software was supported by the Army Research Laboratory Vehicle Technology Center at NASA Glenn Research Center.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076
Price Code: A03

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100
Price Code: A03

ON FLOWFIELD PERIODICITY IN THE NASA TRANSONIC FLUTTER CASCADE, PART II—NUMERICAL STUDY

Rodrick V. Chima, Eric R. McFarland, and Jerry R. Wood
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

and

Jan Lepicovsky
Dynacs Engineering
Brookpark, Ohio 44142

ABSTRACT

The transonic flutter cascade facility at NASA Glenn Research Center was redesigned based on a combined program of experimental measurements and numerical analyses. The objectives of the redesign were to improve the periodicity of the cascade in steady operation, and to better quantify the inlet and exit flow conditions needed for CFD predictions. Part I of this paper describes the experimental measurements, which included static pressure measurements on the blade and endwalls made using both static taps and pressure sensitive paints, cobra probe measurements of the endwall boundary layers and blade wakes, and shadowgraphs of the wave structure. Part II of this paper describes three CFD codes used to analyze the facility, including a multibody panel code, a quasi-three-dimensional viscous code, and a fully three-dimensional viscous code. The measurements and analyses both showed that the operation of the cascade was heavily dependent on the configuration of the sidewalls. Four configurations of the sidewalls were studied and the results are described. For the final configuration, the quasi-three-dimensional viscous code was used to predict the location of mid-passage streamlines for a perfectly periodic cascade. By arranging the tunnel sidewalls to approximate these streamlines, sidewall interference was minimized and excellent periodicity was obtained.

INTRODUCTION

The transonic flutter cascade facility at NASA Glenn Research Center is one of a very few test facilities dedicated to unsteady aerodynamics of oscillating airfoils. The facility combines a transonic linear cascade wind tunnel with a high-speed drive system. The drive system imparts torsional oscillations to the blades at prescribed interblade phase angles and realistic reduced frequencies. Experimental data acquired in this facility serve as benchmark cases for validating unsteady computational fluid dynamic (CFD) codes used to model self-induced cascade flutter.

Lately the facility has been used to study a modern, low aspect ratio fan blade operating near the stall flutter boundary that occurs at high incidence angles and transonic relative Mach numbers. To accurately represent blades oscillating at an interblade phase angle β, it is necessary to have periodicity over at least $360/\beta + 1$ blades. Previous measurements on these blades were reported by Buffum, et al. (1996a, b) at Mach numbers between 0.2 and 0.8, and incidence angles of 0 and 10 degrees. Acceptable periodicity was found over three blades, which was sufficient for measurements of the unsteady pressures at an interblade phase angle of $\beta = 180$ degrees. For smaller interblade phase angles, periodicity was needed over more blades. Adjustments to the boundary layer bleed system improved periodicity for some flow conditions, but were not sufficient for all cases of interest.

Comparisons with various CFD predictions have suggested that the flow incidence angle ahead of the blades was between 0.5 and 1.5 degrees less than the geometric incidence angle between the headboards and the blades (Buffum, et al. 1996a, b.)The present work also suggested that the incidence angle might vary across the face of the cascade. Measured downstream pressures were generally inconsistent with CFD predictions.

A combined experimental and numerical study of the facility was carried out to improve the periodicity of the tunnel, and to better quantify the inlet and exit conditions needed for accurate CFD predictions. Part I of this paper describes the facility in detail and describes a variety of experimental data taken in the facility. The data includes blade and endwall static pressure data, upstream boundary layer data, downstream wake data, and flow visualization measurements made using shadowgraphs and pressure sensitive paints.

Part II of the paper describes several CFD calculations used to understand the original behavior of the facility and to devise improvements to the facility. A panel code was used to analyze the complete tunnel including the sidewalls and nine blades. A quasi-three-dimensional Navier-Stokes code was used to analyze isolated blades under periodic flow conditions. Four configurations of the tunnel sidewalls were analyzed computationally and tested experimentally. It was found that the sidewall configuration had a large impact on the periodicity of the cascade, and that no amount of bleed could correct for poorly configured sidewalls. It was also found that the periodic Navier-Stokes code could be used to determine a sidewall configuration that maximized the periodicity of the cascade.

FACILITY

The transonic flutter cascade facility is shown in figure 1. It consists of an inlet section, a test section with blades, and an exit section connected to a central air exhaust system. The cascade is bounded by *endwalls*, which are analogous to the hub and tip of an annular blade row, and *sidewalls*, which include the headboards, tailboards, and any wall in between. A brief description of the facility is given here for reference, and details are given in Part I of this paper (Lepicovsky, et al. 2000.)

Room air enters the inlet section through a honeycomb in a bellmouth inlet. The headboards upstream of the cascade are adjustable to control the incidence to the cascade. The upstream boundary layers may be bled off through perforations in both the headboards and the endwalls.

The test section has a rectangular cross section 58.6 cm wide by 9.78 cm along the span. Nine blades were located in the test section. The blades were designed and fabricated by Pratt and Whitney. They are similar to a section near the tip of a modern low aspect ratio transonic fan, with an aerodynamic chord of 8.89 cm, a maximum thickness of 4.8 percent chord, a solidity of 1.52, and 60 degrees of stagger. Two blades were instrumented with pressure taps at 15 chordwise locations. Pressure distributions could be measured in most blade passages by moving the instrumented blades to different locations in the cascade.

The blades have constant cross section except near the endwalls where they have large, diamond-shaped fillets. The fillets attach the blades to thick trunnions connected to an external, high-speed drive mechanism that can oscillate the blades at up to 500 Hz. The fillet on the drive-side is larger than the one on the free side. Two blades were instrumented with high frequency response pressure transducers to measure the unsteady response of the oscillating blades. Since the present work concentrated on improving the steady behavior of the cascade, no unsteady measurements will be described here.

The exit section has adjustable tailboards to control the exit flow angle. The tailboards start just ahead of the leading edge of the cascade and can be moved to form scoops that remove the sidewall boundary layers. Downstream of the exit section air is expanded through a diffuser into an exhaust header. The exhaust system is connected to the central air facility at NASA Glenn which maintains a constant exhaust pressure of 30 kPa downstream of a flow control valve.

Three rows of endwall static pressure taps spanned the entire width of the cascade. One row of taps was located 16.5 cm far ahead of the cascade, the second row was located 3.1 cm ahead of the blade leading edges, and the third row was located 8.5 cm downstream of the trailing edges. These three rows of pressure taps were used to assess the periodicity of the cascade flow.

CFD CODES

PCSTAGE

The PCSTAGE turbomachinery analysis panel code developed by McFarland (1993, 1994) was used to model the complete tunnel configuration. The code uses an integral equation solution method to solve the two-dimensional, inviscid flow equations for multiple bodies. Compressibility effects are approximated in the solution. The method is most accurate for low Mach number flows, but can be applied to flows where Mach numbers remain less than one.

A simplified problem that simulated the complete tunnel configuration was developed. Ten bodies were used. Nine of the bodies were the fan blades that make up the cascade section of the tunnel. The tenth body was an elongated blade shape that makes up the tunnel sidewalls. The elongated shape was designed such that the left surface models the right wall of the tunnel and right surface of the body models the left wall.

Key features of the experiment were modeled. The location of the walls with respect to the cascade and the slope of the walls were incorporated into the design of the tenth body. Details such as wall bleed slots and boundary layer scoops were not included. A periodic boundary condition was applied to the entire group of ten bodies. The spacing of the periodic boundary was chosen so that the distance between the left and right tunnel walls was matched by the periodic spacing between the left surface of the tenth body and the right surface of the cascaded image of the body. The use of a single body to represent both tunnel walls eliminates problems of matching tunnel mass flow that occurs when two bodies are used.

The problem was set up to make best use of the PCSTAGE solver. Calculations were made at $M = 0.5$ to minimize compressibility effects. Kutta condition constraints were used on each of the ten bodies rather than assigning a circulation to each. This resulted in an iterative solution, but allowed the solution to determine the flow split around each of cascade bodies.

The problem size was moderate. The elongated body was modeled with 98 panels and the cascade blades with 70 panels each. This resulted in a solution matrix of 748 equations with 748 unknowns. The solution of this matrix provides the surface flow conditions at the center of each body panel. The flow conditions were also calculated at 3147 points in the flow field. These field points plus the surface points were combined using the random points feature of the TecPlot© graphics software to produce the contour plots of the flow field. The PCSTAGE calculation took about 4 minutes on an SGI Indigo 2 workstation.

RVCQ3D

The quasi-three-dimensional (Q-3-D) turbomachinery analysis code RVCQ3D developed by Chima (1987, 1995) was also used to analyze the blades. The code solves the thin-layer Navier-Stokes equations in finite-difference form. Blockage effects can be modeled by specifying a stream sheet thickness that can vary with streamwise distance. Turbulence effects were modeled using the Baldwin-Lomax model, including the original transition model. The flow equations were solved using an explicit Runge-Kutta scheme. A spatially varying time step and implicit residual smoothing were used to accelerate convergence.

A C-type grid was used. The grid had 225 points around the blade and 45 points from the blade to mid-pitch, for a total of 10,125 points.

The spacing at the wall gave $y^+ \approx 3$ at the first grid point. The upstream boundary of the grid was placed at the same location as the near-upstream measurement station used in the experiment, 3.1 cm ahead of the leading edge. The cascade was assumed to be periodic blade-to-blade, so that only one isolated blade was analyzed.

Endwall boundary layer blockage was neglected after early RVCQ3D calculations with 5-10 percent blockage added failed to improve agreement with experimental results. Later three-dimensional calculations also showed that blockage effects were negligible.

Most calculations were run for 1500 iterations, which took about 3.5 minutes on an SGI Indigo 2 workstation. This ensured that exit total pressure was converged to four significant digits.

SWIFT

Three-dimensional viscous analyses of this cascade were run using the multiblock SWIFT code developed by Chima (1996.) A C-type grid was used around the blade, with the same number of points and spacing as used in 2-D. The 3-D grid had 25 points from the endwall to mid-span. The experimental blade has large, diamond-shaped fillets at the walls to support the attachment shafts. In the computations the smaller free-side fillet was modeled, and the flow was assumed to be symmetrical about mid-span. An H-type grid was added upstream, giving a total grid size of 295,424 points.

TEST SECTION CONFIGURATIONS

Four different sidewall configurations were investigated in order to improve the periodicity of the cascade. Three of the configurations are shown in figure 2. The configurations will be referred to by two angles, the headboard angle followed by the tailboard angle, both measured with respect to the horizontal. In each case the blades were positioned at a setting angle of 30 degrees from the horizontal, i.e., staggered 60 degrees.

Original Configuration (20/30)

In the original configuration the headboard was set at 20 degrees, giving a geometric incidence angle of $i_{fl} = 10$ degrees. The tailboard was positioned at the blade setting angle of 30 degrees. This was one of the configurations described by Buffum, et al. (1996a, b.)

Figure 3 compares the measured surface pressure distribution at $M = 0.8$ with distributions calculated using RVCQ3D at several incidence angles i_{fl}. Here the pressure coefficient is given by

$$C_p = \frac{p - p_{in}}{\frac{1}{2}\rho_{in} V_{in}^2}. \qquad (1)$$

The best agreement with the data was for i_{fl} between 8 and 9 degrees, indicating that the cascade was operating at an effective incidence angle that was less than the 10 degree angle between the headboards and the blades. This is consistent with the incidence angle corrections mentioned by Buffum, et al. (1996b.) In the present study, cobra probe measurements of the upstream flow also showed that the incidence angle was about 8.5 degrees with no bleed flow.

Figure 4 shows static pressures $p/p_{0,in}$ measured at a nominal Mach number $M = 0.8$ along the three rows of endwall static taps shown in figure 1. Figure 4a shows measurements from the original configuration. Far upstream the pressures were fairly uniform. Near upstream there was a large variation in static pressure across the cascade, corresponding to Mach number variations between 0.68 and 0.85. Downstream the pressure varied about as much as near upstream. Adjustments to the sidewall and endwall bleed valves had local effects on the pressure distributions but did not significantly improve the overall uniformity of the flow. All bleed valves were closed in the subsequent work.

The inlet Mach number cannot be specified directly in RVCQ3D. Instead, $p_{exit}/p_{0,in}$ is specified and the inlet Mach number is computed as part of the solution. The measurements in figure 4a give a nominal static pressure ratio $p_{exit}/p_{in} = 1.124$ across the cascade, corresponding to $p_{exit}/p_{0,in} = 0.744$ for $M = 0.8$. Using the measured pressure ratio, RVCQ3D gave an inlet Mach number that was much too low. It was necessary to drop the exit pressure ratio to 0.70 to recover the correct inlet Mach number, giving $p_{exit}/p_{in} = 1.067$.

Figure 5 shows Mach number contours calculated for this configuration using PCSTAGE. Figure 5a shows calculations of the original configuration. The calculations show a large variation in Mach number across the entire face of the cascade. Only three passages near the center of the cascade see the nominal Mach number shown by orange contours. The largest variation in Mach number occurs at the bend in the left sidewall, where the measured pressure was lowest (figure 4a.) Adjustments of the sidewall scoop in this area modified the pressures locally but failed to improve the large variations ahead of the cascade.

There was some speculation that three-dimensional effects due to endwall boundary layers, blade fillets, and the large suction surface separation might be important in this cascade. Limited measurements by Buffum, et al. (1996a, b) showed that surface pressures were nearly uniform along the span, with minor variations at 17.5 percent span. Blade and endwall pressure contours calculated with the SWIFT code at $M = 0.8$ and $i_{fl} = 8°$ are shown in figure 6. Here the flow is separated from the leading edge to about 45 percent chord, but the surface pressures are nearly uniform except near the endwalls. Three-dimensional effects were thus seen to be minimal and were neglected in subsequent work.

Original Configuration with Blades Removed

PCSTAGE calculations of the original configuration showed that the sidewalls were heavily loaded while the blades were lightly loaded, indicating that the sidewalls were doing most of the turning. To verify this observation the tunnel was run with all of the blades removed. The measured pressure distributions in figure 4b and the PCSTAGE calculations in figure 5b are very similar to the corresponding results with the blades in place (4a and 5a.) These results showed that most of the flow variation in the original tunnel configuration was due to the turning of the sidewalls.

First Redesign (21.5/24.5)

To improve the periodicity of the cascade it was thus necessary to match the sidewall turning to that of a perfectly periodic cascade. For the first redesign the sidewall turning angle was chosen to match the turning predicted by RVCQ3D. Calculations were made over a range of Mach numbers for incidence angles of 8 and 10 degrees. The calculated turning angle is plotted versus inlet Mach number in figure 7. For both incidence angles and subsonic speeds the flow on the suction surface separates at the leading edge and reattaches at about 49 percent chord, as determined from the sign of the axial velocity component one grid point off the blade. Because of the large separation bubble the flow does not turn to the blade setting angle, and the resulting turning is less than the geometric turning of 8 or 10 degrees.

For eight degrees incidence the predicted turning varies from 3.5 degrees at low speeds to 3 degrees at $M = 1$. For $M > 1$ a supersonic expansion around the leading edge keeps the flow attached until the supersonic region is terminated by a shock. Thus the separation is reduced and the turning is increased slightly. For $M > 1.2$ the calculations become completely supersonic on the upstream boundary and it is impossible to set the flow angle.

For ten degrees incidence the turning varies continuously from about 5 degrees at low speeds to 4 degrees at $M \approx 1.2$. At this higher incidence the flow separates at the leading edge for all Mach numbers.

For the first redesign the headboards were set at 21.5 degrees (8.5 degrees incidence) since CFD calculations had suggested that previous data was effectively at that angle. The tailboards were set at 24.5 degrees (three degrees of turning.) These sideboard angles were expected to approximate the flow turning for a periodic cascade over a wide range of flow speeds.

Figure 5c shows Mach number contours calculated for this configuration using PCSTAGE. The orange contour shows fairly uniform flow ahead of six of the of the interior passages. Two minor problems with this configuration are also evident. First, the passage between the left sidewall and first blade converges and could choke. Second, the passage between the right sidewall and last blade diverges and could separate.

Figure 4c shows that the measured endwall pressures were considerably more uniform than the original configuration (4a.) The near-upstream pressures were nonuniform at the ends of the cascade, probably because of the two problems noted above.

Experimental results showed that the cascade could no longer reach supersonic conditions in this configuration. A one-dimensional analysis showed that the original configuration had its minimum area or throat in the inlet section. In the first redesign the headboard angle was reduced 1.5 degrees. This moved the throat inside the blade row so that the blade row choked before the upstream flow could become supersonic.

Second Redesign (20/24)

The second redesign was an attempt to minimize interference in the left-most and right-most passages, and to restore supersonic operation of the cascade. Interference was minimized by shaping the sidewalls like a mid-passage streamline. Supersonic operation was restored by returning the headboard angle to 20 degrees.

In the previous designs the sidewalls were spaced one blade pitch from their neighboring blades. With this spacing the sidewalls approximate the stagnation streamline, including the blade surfaces. Since the original sidewalls were straight and met near the leading edge of the cascade (see figure 2), they made a poor approximation of the stagnation streamline. Computed particle traces were used to find a better approximation to a passage streamline.

The blades were analyzed with RVCQ3D at $M = 0.8$ and $i_{fl} = 10°$. Particle traces for that solution (figure 8) showed that the streamlines near the center of a blade passage could be approximated with three line segments: one upstream at the nominal inflow angle, one downstream at the calculated turning angle, and a third within the passage at the blade setting angle. Thus, for the second redesign the headboard angle was returned to 20 degrees to restore supersonic operation, the tailboard angle was set to 24 degrees to match the turning predicted by RVCQ3D for 10 degrees of incidence, and a straight, 30 degree insert was fabricated to connect the two. The sidewalls were spaced one-half pitch from their neighboring blade.

Figure 5d shows Mach number contours calculated for this configuration using PCSTAGE. The contours show very uniform flow ahead of the cascade. They also show that the addition of a third wall segment between the leading and trailing edges has nearly eliminated the problems with the passages nearest the walls.

Figure 4d shows that the measured endwall pressures were nearly uniform. The static pressure ratio across the cascade is 1.058, corresponding to $p_{exit}/p_{0,in} = 0.70$ for $M = 0.8$. This was exactly the pressure ratio required by RVCQ3D to recover the correct inlet Mach number.

Figures 9a and 9b show calculations made with RVCQ3D at $M = 0.8$ and $i_{fl} = 9°$. Mach contours in figure 9a show that the flow remains entirely subsonic. The suction surface flow separates at the leading edge and reattaches at 49 percent chord. The boundary layer remains very thick even into the wake. Figure 9b compares computed surface pressure coefficients with data measured after the second redesign. The best comparison was found for $i_{fl} = 9°$. Preliminary measurements show that the incidence angle varies between nine and ten degrees.

The calculations agree closely with the measurements on the pressure side. The flat pressure distribution on the suction side from zero to 20 percent chord is characteristic of separated flow and is very sensitive to the turbulence model used. Consequently, this may prove to be an unrealistic operating point for linearized Euler calculations sometimes used for unsteady flutter analysis.

Figures 10a and 10b show calculations made with RVCQ3D at $M = 1.0$ and $i_{fl} = 8°$. Mach contours in figure 10a show that the flow expands around the leading edge to $M \approx 1.15$, then decelerates through a normal shock. Wall pressure contours computed with the three-dimensional SWIFT code (figure 6) show a similar shock structure. Figure 10b compares computed surface pressure coefficients with measured data. Here the best comparison was for $i_{fl} = 8°$. Although the flowfield at $M = 1.0$ (figure 10a) is quite different from the flowfield at $M = 0.8$ (figure 9a), the surface pressure distributions (figures 9b and 10b) are similar.

CONCLUDING REMARKS

The transonic flutter cascade facility at NASA Glenn Research Center is a unique facility for measuring unsteady aerodynamics of oscillating turbomachinery blades. Recent measurements in the facility have concentrated on the stall-flutter boundary of a modern fan section at high incidence angles and transonic Mach numbers. Previous unsteady measurements have been limited to 180 degrees interblade phase angle due to poor cascade periodicity. CFD calculations have also revealed inconsistencies between measured flow conditions and the boundary conditions that gave the best agreement with the data.

A combined experimental and numerical study was carried out to improve the periodicity of the facility and to better quantify the boundary conditions for CFD calculations. The experimental part of this work is described in Part I of this paper (Lepicovsky et al. 2000.) The numerical work is described here.

Calculations were made using three CFD codes. The PCSTAGE panel code was used to analyze the complete test section, including the sidewalls and all nine blades. The RVCQ3D quasi-three-dimensional viscous code was used to analyze the isolated blades. The SWIFT three-dimensional viscous code was used to investigate 3-D effects of fillets, endwall blockage, and separation.

Four configurations of the tunnel sidewalls were analyzed and tested:

1 In the original configuration the headboards were set for 10 degrees geometric incidence and the tailboards were positioned at the setting angle of the blades. Several measurement techniques showed that the flow through the cascade was nonuniform. Data from three arrays of endwall pressure taps parallel to the cascade face gave the quickest indication of flow nonuniformity. PCSTAGE calculations showed that the nonuniformities were caused by the sidewall configuration. RVCQ3D calculations and cobra probe measurements showed that the cascade was operating at a lower effective incidence than expected. SWIFT calculations showed that three-dimensional effects were negligible.

2 The original configuration was also tested and analyzed without the blades. PCSTAGE calculations and the experimental data both showed that the nonuniformities in the flow were due to the turning of the sidewalls and not to the blades. Discrepancies in previous data from the facility were thus due to tunnel wall interference at the high incidence angles of interest for the stall-flutter problem.

3 In the first redesign the headboards were set at the effective incidence angle suggested by CFD calculations. The tailboards were set to the exit angle predicted by RVCQ3D. PCSTAGE predictions and endwall pressure measurements showed considerable improvement in flow uniformity, except near the sidewalls. Experimental results showed that the cascade could no longer reach supersonic speeds in this configuration. A one-dimensional analysis showed that the reduced headboard angle caused the facility to choke in the cascade before supersonic conditions could be reached upstream.

4 In the second redesign the headboards were returned to 10 degrees incidence to restore supersonic operation, and the tailboards were set to the exit angle predicted by RVCQ3D. An insert was fabricated to connect the head and tailboards, and the sideboards were moved laterally one-half pitch to approximate the mid-passage streamlines predicted by RVCQ3D. PCSTAGE calculations and endwall pressure measurements showed nearly uniform conditions across the entire face of the cascade. RVCQ3D calculations showed very good agreement with blade surface pressures and cascade pressure ratios measured in this configuration.

The results showed how a periodic viscous analysis code like RVCQ3D can be used to minimize sidewall interference in any cascade facility. First, calculated blade turning angles can be used to set headboard and tailboard angles. Second, calculated streamlines can be used for sidewall shapes. Sidewalls should probably be located at mid-passage since mid-passage streamlines are smoother than the stagnation streamline and have less influence on the blade loading.

Now that acceptable periodicity has been obtained in the NASA transonic flutter cascade facility, future work will concentrate on obtaining unsteady data at several interblade phase angles. The sidewall insert should be sufficient for all Mach numbers of interest, but it will have to be modified for other incidence angles or stagger angles.

Continuous collaboration between the experimental and computational parts of this work proved helpful to everyone involved. The computational results helped to explain the operation of the tunnel, helped to eliminate three-dimensional effects as a potential cause of problems, and guided the placement of the sidewalls. The experimental results have helped to validate steady CFD models, and eventually will help to validate unsteady models for transonic stall-flutter.

REFERENCES

Buffum, D. H., Capece, V. R., King, A. J., and EL-Aini, Y. M., 1996a, "Oscillating Cascade Aerodynamics at Large Mean Incidence," ASME Paper 96-GT-339. Also NASA TM-107247.

Buffum, D. H., Capece, V. R., King, A. J., and EL-Aini, Y. M., 1996b, "Experimental Investigation of Unsteady Flows at Large Incidence Angles in a Linear Oscillating Cascade," AIAA Paper 96-2823. Also NASA TM-107283.

Chima, R. V., 1987, "Explicit Multigrid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbomachinery," *AIAA J. Propulsion and Power*, Vol. 3, No. 5, pp. 397-405.

Chima, R. V., 1995, "A k-ω Turbulence Model for Quasi-Three-Dimensional Turbomachinery Flows," AIAA Paper 96-0248. Also NASA TM-107051.

Chima, Rodrick V., 1996, "Calculation of Tip Clearance Effects in a Transonic Compressor Rotor," ASME Paper 96-GT-114. Also NASA TM-107216.

Lepicovsky, J., McFarland, E. R., Chima, R. V., and Wood, J. R., 2000, "On Flowfield Periodicity in the NASA Transonic Flutter Cascade, Part I - Experimental Study," ASME Paper 2000-GT-0572.

McFarland, E. R., 1993, "An Integral Equation Solution for Multistage Turbomachinery Design Calculations," ASME paper 93-GT-41.

McFarland E. R., 1994, "Use of Preliminary Design Methods in the Analysis of Multi-Stage Turbomachinery," NASA CP 3282, Vol. 2.

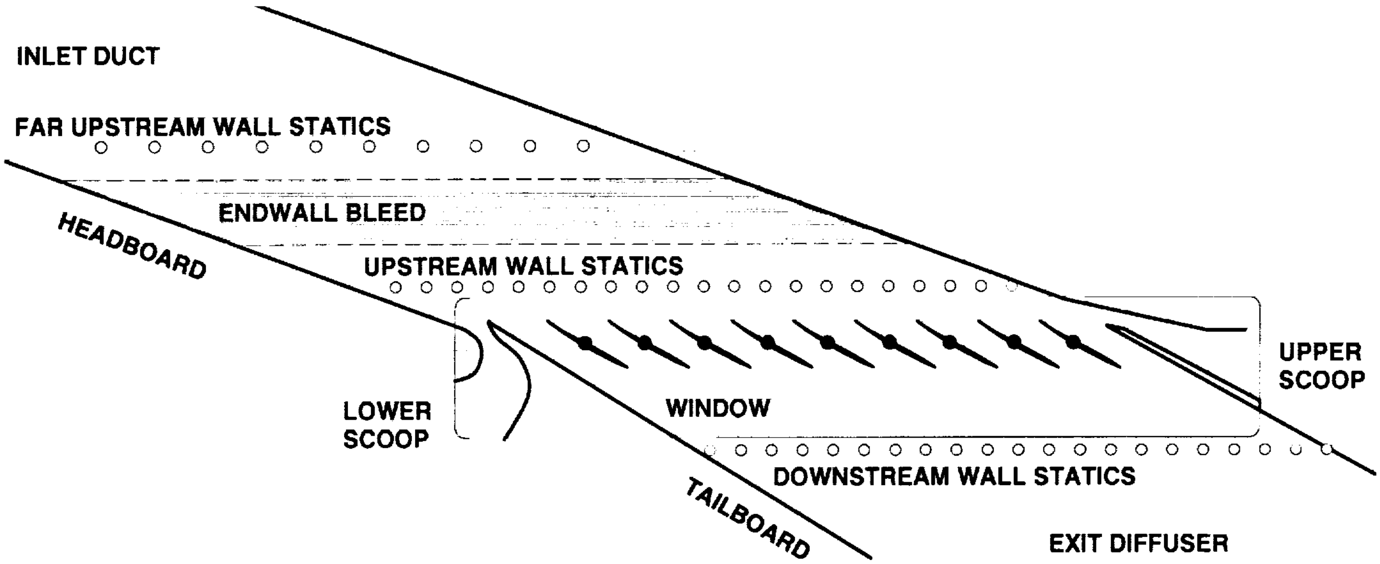

Figure 1 — NASA Glenn Research Center oscillating cascade facility.

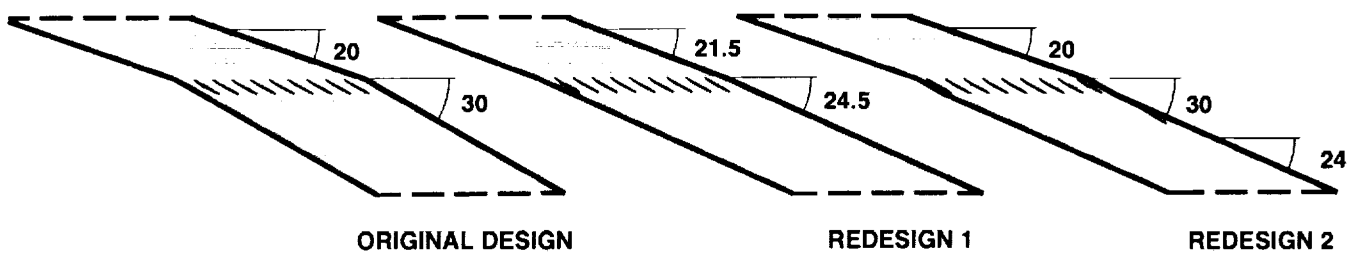

Figure 2 — Three tunnel configurations studied.

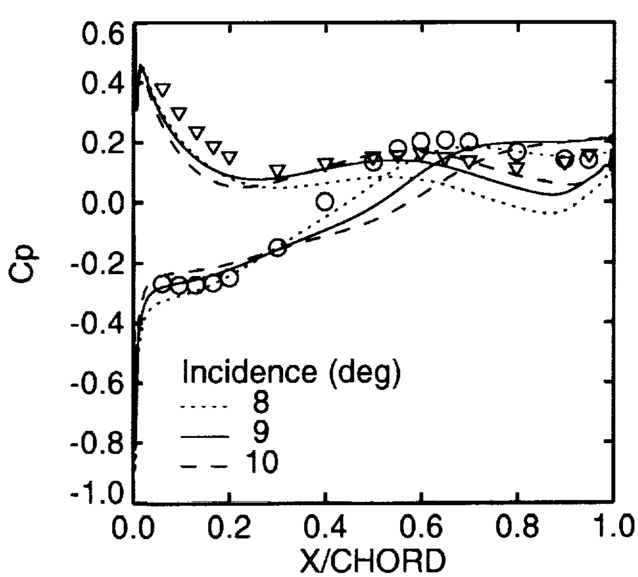

Figure 3 — Effect of incidence angle i_{fl} on pressure coefficient computed with RVCQ3D

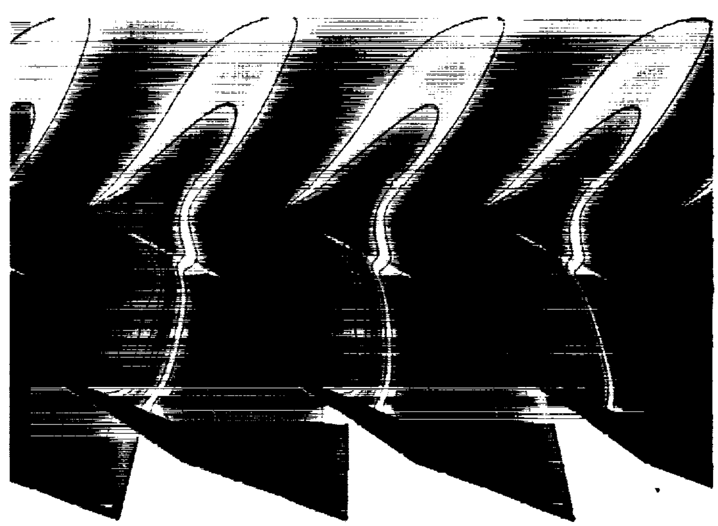

Figure 6 — Pressure contours on the blade and endwall computed with the SWIFT code, $M = 1.0$, $i_{fl} = 8°$.

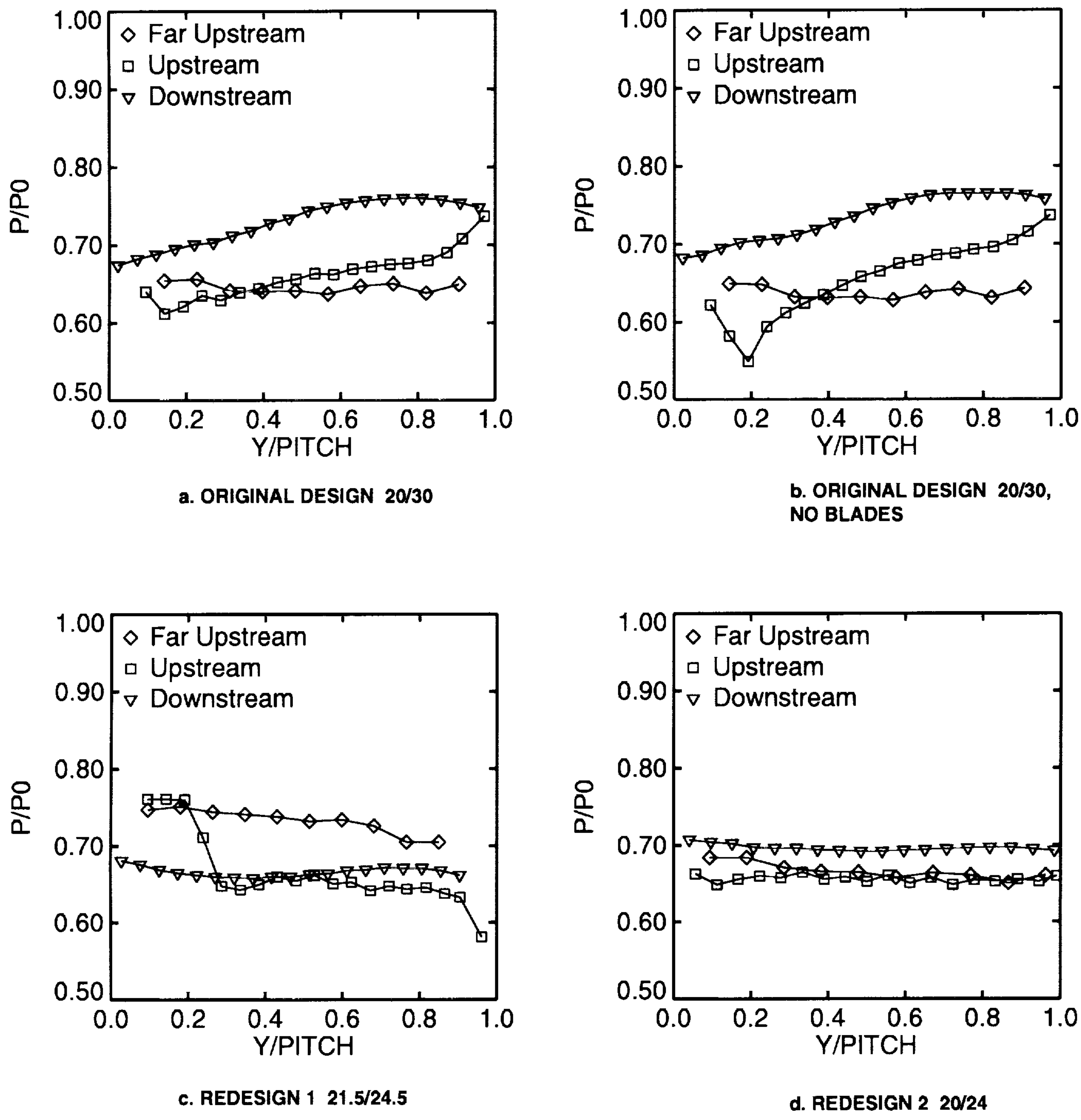

Figure 4 — Measured sidewall pressures, $M = 0.8$.

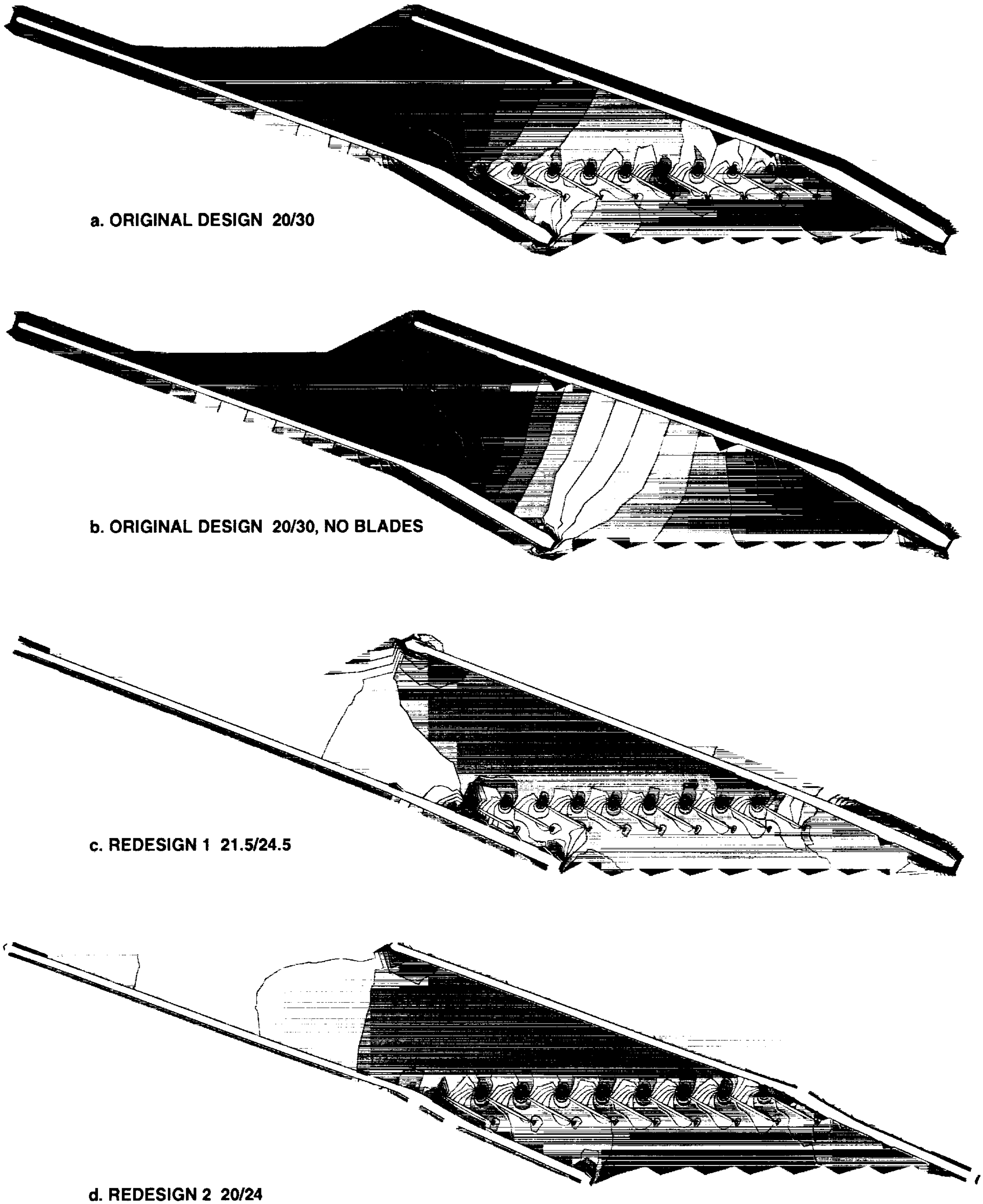

Figure 5 — Mach number contours computed with PCSTAGE, contour increment = 0.02

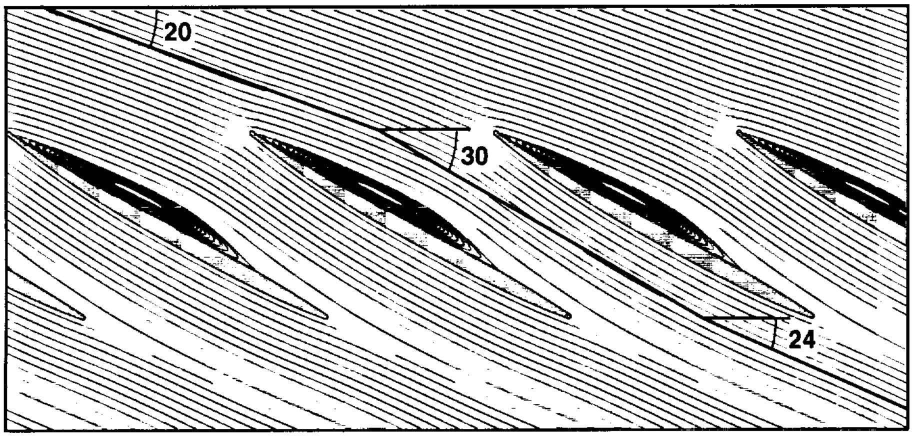

Figure 8 — Particle traces computed with RVCQ3D, $M = 0.8$, $i_{fl} = 10°$.

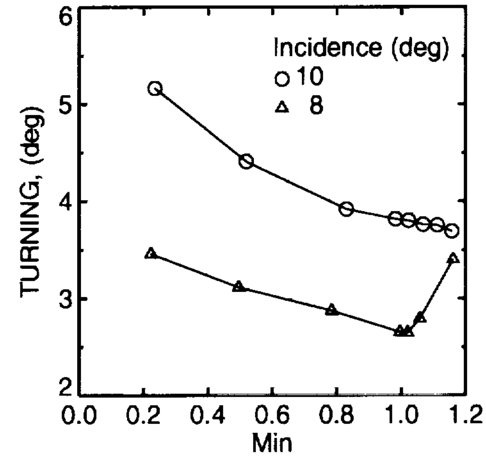

Figure 7 — Cascade turning angle vs. Mach no. computed with RVCQ3D.

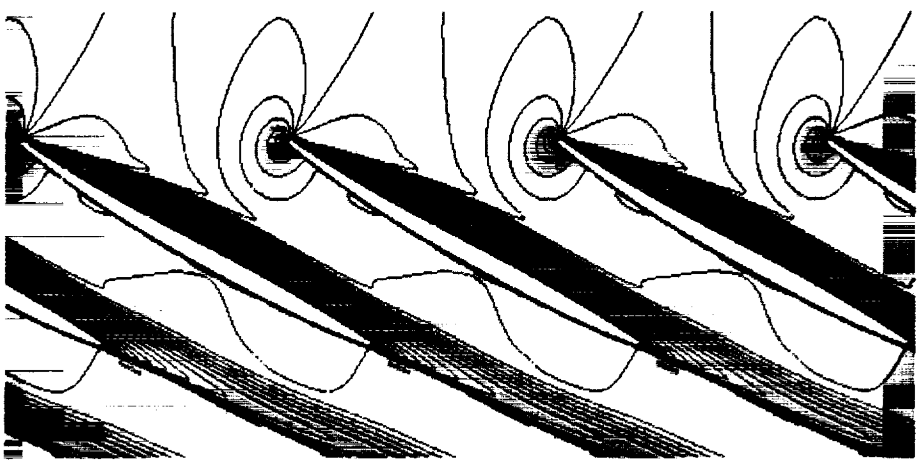

Figure 9a — Mach contours computed with RVCQ3D, $M = 0.8$, $i_{fl} = 9°$, contour increment = 0.05.

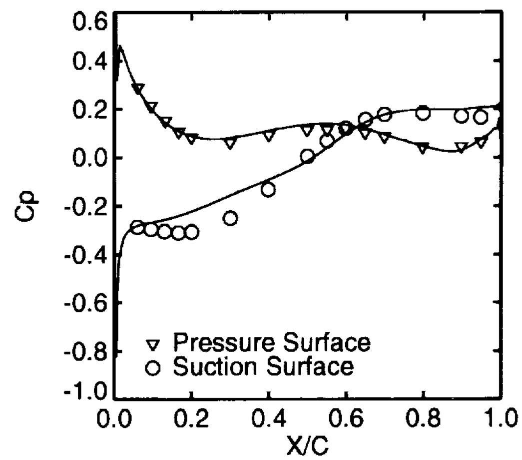

Figure 9b — Comparison of computed and measured pressure coefficient, second redesign, $M = 0.8$, $i_{fl} = 9°$.

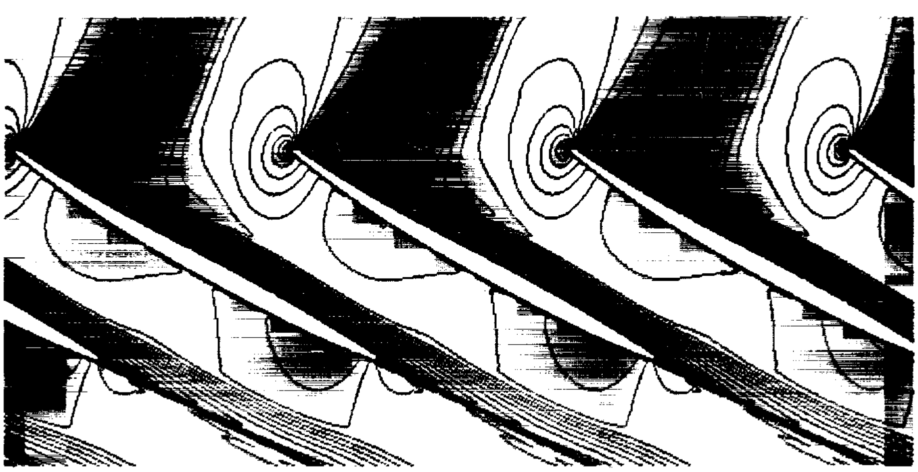

Figure 10a — Mach contours computed with RVCQ3D, $M = 1.0$, $i_{fl} = 8°$, contour increment = 0.05, heavy line = 1.0.

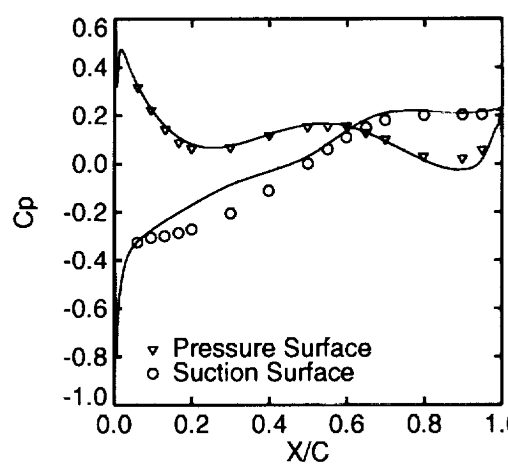

Figure 10b — Comparison of computed and measured pressure coefficient, second redesign, $M = 1.0$, $i_{fl} = 8°$.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
March 2000

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
On Flowfield Periodicity in the NASA Transonic Flutter Cascade, Part II—Numerical Study

5. FUNDING NUMBERS
WU-523-26-13-00

6. AUTHOR(S)
Rodrick V. Chima, Eric R. McFarland, Jerry R. Wood, and Jan Lepicovsky

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
John H. Glenn Research Center at Lewis Field
Cleveland, Ohio 44135-3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E-12177

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM—2000-209933
ASME 2000-GT-0573

11. SUPPLEMENTARY NOTES
Prepared for the 45th International Gas Turbine and Aeroengine Technical Congress sponsored by the American Society of Mechanical Engineers, Munich, Germany, May 8–11, 2000. RodrickV. Chima, Eric R. McFarland, and Jerry R. Wood, NASA Glenn Research Center; and Jan Lepicovsky, Dynacs Engineering Company, Inc., 2001 Aerospace Parkway, Brook Park, Ohio 44142. Responsible person, Rodrick V. Chima, organization code 5810, (216) 433-5919.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category: 07 Distribution: Nonstandard

This publication is available from the NASA Center for AeroSpace Information, (301) 621-0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

The transonic flutter cascade facility at NASA Glenn Research Center was redesigned based on a combined program of experimental measurements and numerical analyses. The objectives of the redesign were to improve the periodicity of the cascade in steady operation, and to better quantify the inlet and exit flow conditions needed for CFD predictions. Part I of this paper describes the experimental measurements, which included static pressure measurements on the blade and endwalls made using both static taps and pressure sensitive paints, cobra probe measurements of the endwall boundary layers and blade wakes, and shadowgraphs of the wave structure. Part II of this paper describes three CFD codes used to analyze the facility, including a multibody panel code, a quasi-three-dimensional viscous code, and a fully three-dimensional viscous code. The measurements and analyses both showed that the operation of the cascade was heavily dependent on the configuration of the sidewalls. Four configurations of the sidewalls were studied and the results are described. For the final configuration, the quasi-three-dimensional viscous code was used to predict the location of mid-passage streamlines for a perfectly periodic cascade. By arranging the tunnel sidewalls to approximate these streamlines, sidewall interference was minimized and excellent periodicity was obtained.

14. SUBJECT TERMS
Compressors; Cascades; Computational fluid dynamics; Turbomachinery; Fans; CFD; Transonic flow; Flutter

15. NUMBER OF PAGES
15

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	